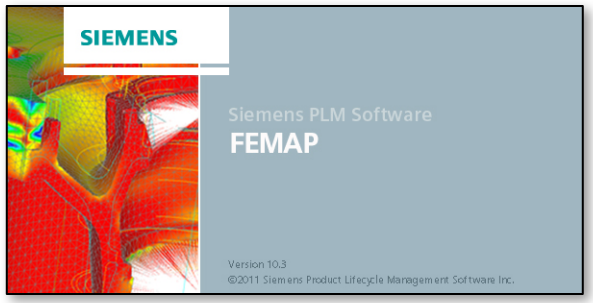
FINITE ELEMENT ANALYSIS

Predictive Engineering

Foundations of FEA Modeling with Femap and NX Nastran

A broad introduction to modern finite element analysis and modeling techniques using Femap and NX Nastran.



2011 – All Rights Reserved

Foundations of FEA Modeling with Femap and NX Nastran

Day 1 Course Outline:

I. Finite Element Technology - Basics

- a.) The concept of finite element analysis - nodes, DOF, elements
- b.) Basic element types - a quick overview
- c.) Linear, elastic FEA
- d.) F = K*U
- e.) Workshop I: Introduction to Femap and NX Nastran
- f) Femap Productivity Notes

II. Finite Element Technology – Beam and Isoparametric Elements

- a.) Beam Elements:
  - i.) Theory
  - ii.) Workshop II: Introduction to Beam Elements
- b.) Isoparametric Solid Elements:
  - i.) Theory
  - ii.) Workshop III: Element Quality
- c.) Isoparametric Plate Elements:
  - i.) Theory
  - ii.) Workshop IV: Introduction to Plate Elements
  - iii.) Workshop V: Mesh Refinement

2011 – All Rights Reserved

Day 1 Course Outline (continued):

III. Foundations

a.) Units and Managing Model Visibility:

i.) Background on FEA Units Usage

ii.) Workshop VI: Blanking, Grouping and Visibility

Day 2 Course Outline

IV. Constraints and Loads Modeling

a.) Engineering Assessment of Constraints

i.) Theory

ii.) Workshop VII: Constraints, Loads and Non-Manifold Geometry

V. Assembly Modeling Basics

a.) Nastran RBE2 and RBE3 Elements

i.) Theory

ii.) Workshop VIII: Nastran Rigid Links (RBE2) versus Nastran Force Interpolation (RBE3)

b.) Application of RBE Elements

i.) Workshop IX: Tying Together Different Element Types.

2011 – All Rights Reserved

Day 2 Course Outline (continued):

VI. Assembly Modeling Basics

a.) Idealization of Engineering Systems

i.) Workshop X: Introduction to Normal Modes Analysis

VII. Results Validation

a.) Understanding Stress Results

i.) Theory of von Mises versus principal stresses

ii.) Workshop XI: Understanding of Von Mises versus Principal versus Transformed

iii.) Workshop XII: Understanding Averaging and Contouring Stress Data

VIII. Fatigue

a.) Concepts of Basic Fatigue in Metal Materials

IX. Extra Material

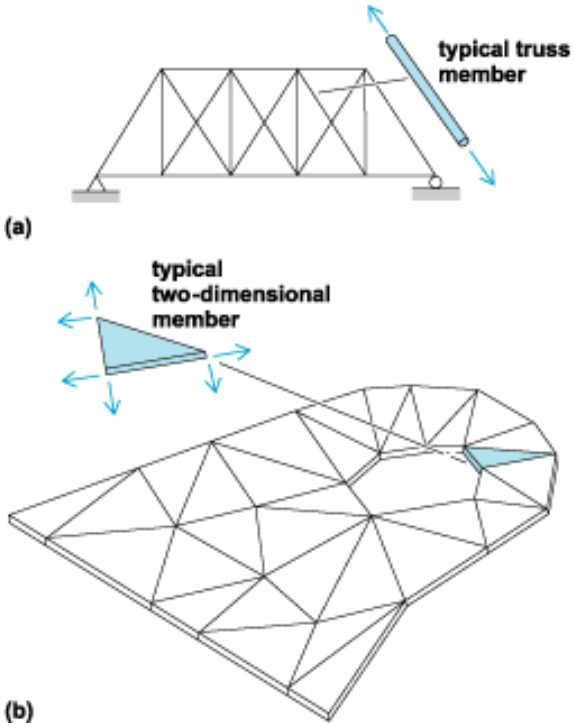
2011 – All Rights Reserved

Finite Element Analysis:

A numerical analysis technique for obtaining approximate solutions to many types of engineering problems. The need for numerical methods arises from the fact that for most practical engineering problems analytical solutions do not exist. While the governing equations and boundary conditions can usually be written for these problems, difficulties introduced by either irregular geometry or other discontinuities render the problems intractable analytically. To obtain a solution, the engineer must make simplifying assumptions, reducing the problem to one that can be solved, or a numerical procedure must be used. In an analytic solution, the unknown quantity is given by a mathematical function valid at an infinite number of locations in the region under study, while numerical methods provide approximate values of the unknown quantity only at discrete points in the region. In the finite element method, the region of interest is divided up into numerous connected sub-regions or elements within which approximate functions (usually polynomials) are used to represent the unknown quantity.

The physical concept on which the finite element method is based has its origins in the theory of structures. The idea of building up a structure by fitting together a number of structural elements (see illustration) was used in the early truss and framework analysis approaches employed in the design of bridges and buildings in the early 1900s. By knowing the characteristics of individual structural elements and combining them, the governing equations for the entire structure could be obtained. This process produces a set of simultaneous algebraic equations. The limitation on the number of equations that could be solved posed a severe restriction on the analysis. The introduction of the digital computer has made possible the solution of the large-order systems of equations.

The finite element method is one of the most powerful approaches for approximate solutions to a wide range of problems in mathematical physics. The method has achieved acceptance in nearly every branch of engineering and is the preferred approach in structural mechanics and heat transfer. Its application has extended to soil mechanics, heat transfer, fluid flow, magnetic field calculations, and other areas.



Structures modeled by fitting together structural elements: (a) truss structure; (b) two-dimensional planar structure.

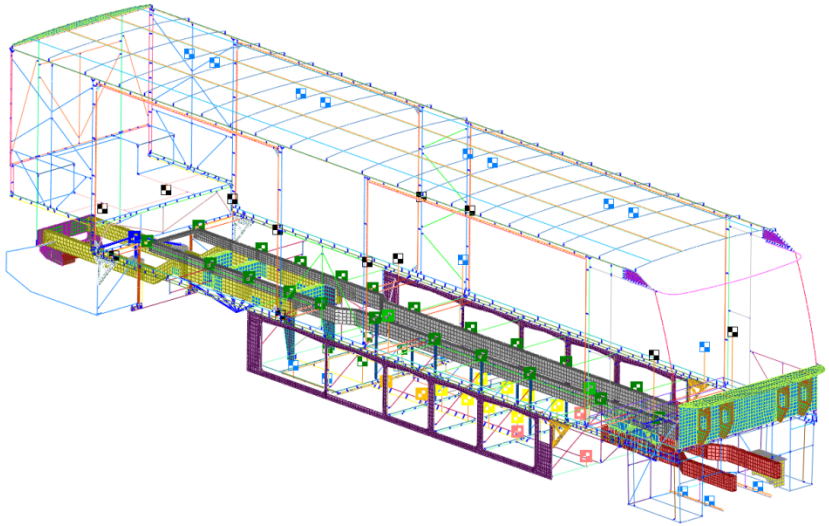
From McGraw-Hill Science and Technology Encyclopedia, 5th Ed.

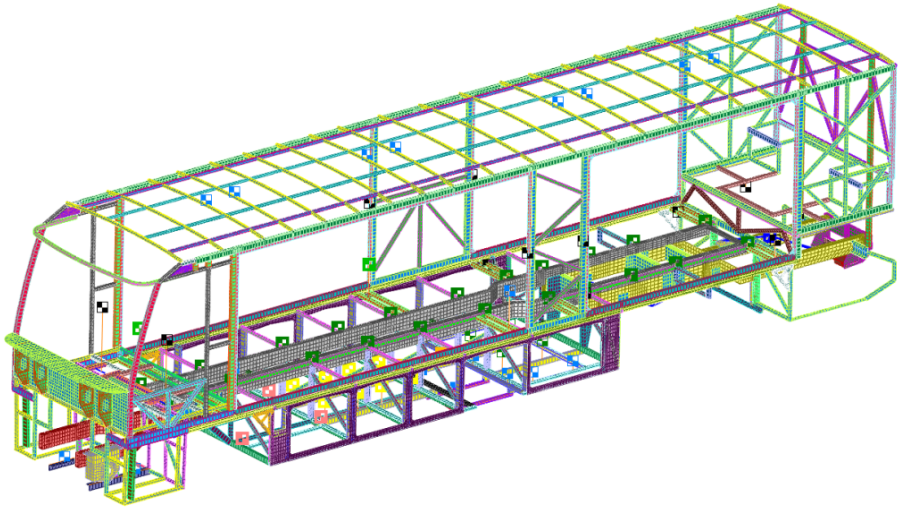
2011 – All Rights Reserved

Foundations of FEA Modeling with Femap and NX Nastran

Four things to consider about FEA:

- Engineering assessment of loads.
- Visualization of structural constraints into modeling constraints.
- Idealization of geometry into a numerical model
- Discretization of continua into a finite element analysis grid.





A dominate contribution to the inaccuracy of a FEA model is loads.

2011 – All Rights Reserved

Finite Element Analysis Concepts

Nodes are used to define the geometry of the finite element (that is to say "its spatial characteristics"). Nodes have degrees-of-freedom and can translate (3 DOF (TX, TY, & TZ)) and rotate (3 DOF (RX, RY, & RZ)) in space.

Finite elements can be classed as point, line, surface and solid elements. Another way to think of these elements is as having 0-D, 1-D, 2-D and 3-D characteristics (D=dimensional).

- 0-D elements are created on one node and can be meshed on geometric points.
- 1-D elements are created on two nodes and can be meshed on geometric lines.
- 2-D elements are created on three or four nodes (triangular or quad) and can be meshed on geometric surfaces.
- 3-D elements are created on a minimum of four nodes (tetrahedral) or eight nodes (brick or hexahedral) and can be meshed on geometric solids.

Examples of various element types are:

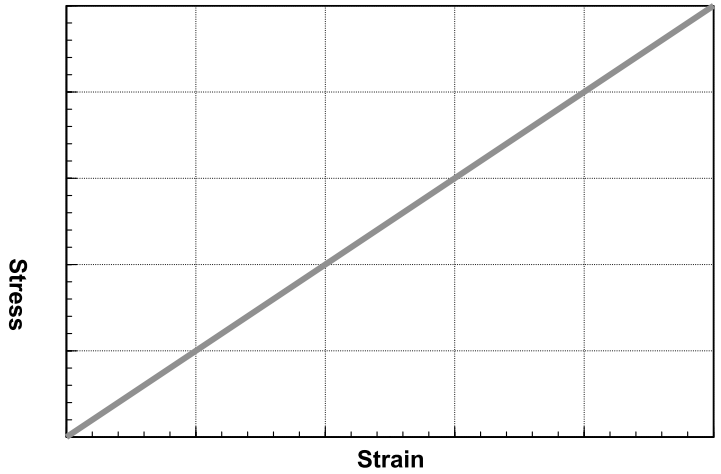
- 0-D elements are mass elements used to simulate concentrated weight without stiffness.
- 1-D elements are beam elements used to model space-frame structures (e.g., bus frames).
- 2-D elements are plate elements used to model thin walled structures (e.g., pressure vessels, airplane skins, sheet metal, ships or structural steel framing).
- 3-D elements are solid elements used to model thick, contoured objects (e.g., castings).

2011 – All Rights Reserved

Finite Element Technology

Linear, Elastic, Static Analysis (99% of the world)

- Stresses can be scaled as a linear function of the loads.
- The structure is elastic.
- Displacements can be scaled as a ratio of elastic moduli.



$$\sigma = \frac{\mathbf{Force}}{\mathbf{Area}}$$

Stress is independent of your material choice.

Application: If the load is a force/pressure, then the resulting stress is just a function of geometry and not the material (i.e., homogeneous materials).

$$\sum \mathbf{F} = 0$$

Static means no acceleration

Application: Your structure must be constrained in all six DOF's (translation and rotation). If it is not constrained correctly – it can't be solved.

2011 – All Rights Reserved

Finite Element Technology

The following section describes how the finite element method works. It shows how basic mechanics are used to generate the finite element displacements and stresses from a structure.

FEA is based on the displacement method, which boils down to:

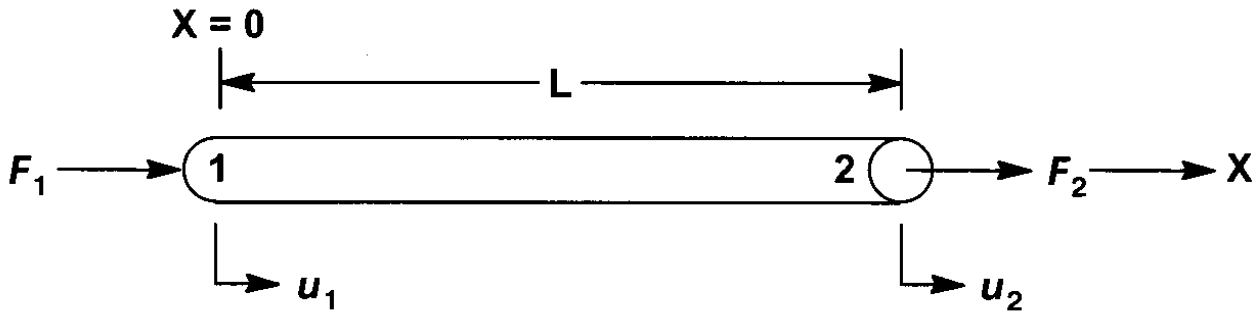
$$\{\mathbf{F}\} = [\mathbf{K}]\{\mathbf{u}\}$$

Typically, one knows something about the loads (F) that are applied to the structure and likewise its stiffness (K). The unknowns are the displacements (u) within the structure after a load is applied. Hence, the method inverts the stiffness matrix and solves for the displacements. With displacements, one can calculate strains and with strains you have stresses and so on and so forth.

A simple rod example is provided showing how this process works.

2011 – All Rights Reserved

Finite Element Technology



Step 1: Satisfy static equilibrium

$$\sum \mathbf{F_x} = \mathbf{F_1} + \mathbf{F_2} = 0$$

$$\mathbf{F_1} = -\mathbf{F_2}$$

Step 2: Relate strain to displacements

$$\varepsilon_x = \frac{\Delta L}{L} = \frac{u_2 - u_1}{L}$$

Step 3: Relate stress to strain

$$\sigma_x = E\varepsilon_x$$

2011 – All Rights Reserved

Finite Element Technology

Step 4: Relate force to stress

$$\sigma_{x1} = -\frac{F_1}{A} \quad \text{and} \quad \sigma_{x2} = \frac{F_2}{A}$$

The minus sign is required since a positive tensile stress at End 1 is in the negative x direction.

Step 5: Relate force to displacement

Using the prior equations and performing a little substitution yields:

$$-F_1 = \sigma_x A = E\varepsilon_x A = \frac{EA}{L}(u_2 - u_1)$$

$$-F_1 = \frac{EA}{L}u_2 - \frac{EA}{L}u_1$$

Step 6: Assemble matrix

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$

which give us: $\{F\} = [K]\{u\}$

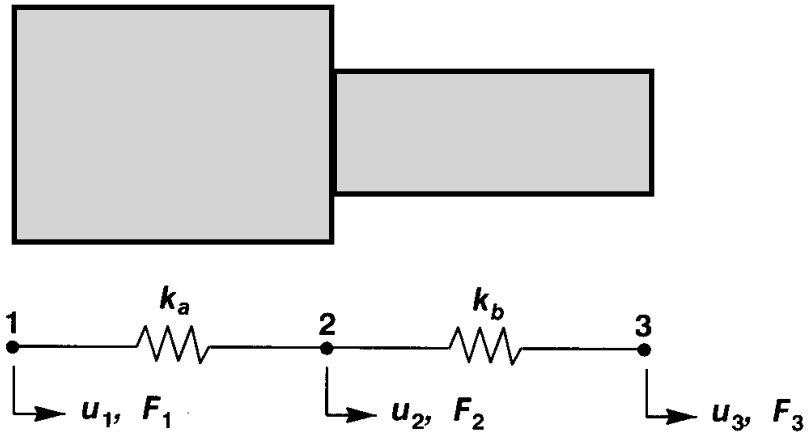
* If u_1 and u_2 are non-zero then an infinite number of solutions are possible or in mathematical terms, the determinant of the stiffness matrix "K" is singular. So, although we have the matrix terms, we don't have a FEA solution.

2011 – All Rights Reserved

Finite Element Technology

This example demonstrates how the FEA method works and illustrates one of the most common errors that new users encounter when they submit their models for analysis.

On the right, we start with a two spring model that has one DOF at each "node". We develop the matrix and one will note that with three nodes we have a matrix that is 3x3.



$$k_a = \frac{E_a A_a}{L_a} = \frac{1 \bullet 2}{1} = 2 \qquad \mathbf{k_b} = \frac{\mathbf{E_b A_b}}{\mathbf{L_b}} = \frac{1 \bullet 1}{1} = 1$$

$$\begin{Bmatrix} \mathbf{F}_1 \\ \mathbf{F}_2 \\ \mathbf{F}_3 \end{Bmatrix} = \begin{bmatrix} 2 & -2 & 0 \\ -2 & 2+1 & -1 \\ 0 & -1 & 1 \end{bmatrix} \begin{Bmatrix} \mathbf{u}_1 \\ \mathbf{u}_2 \\ \mathbf{u}_3 \end{Bmatrix}$$

*Where nodes share elements, they share stiffness terms. Off diagonal terms are zero. For a very large matrix the majority of the terms are zero. Hence, the name for NX Nastran's default solver "Sparse Matrix".

2011 – All Rights Reserved

Finite Element Technology

$$\begin{Bmatrix} \mathbf{F}_1 \\ \mathbf{F}_2 \\ \mathbf{F}_3 \end{Bmatrix} = \begin{bmatrix} 2 & -2 & 0 \\ -2 & 2+1 & -1 \\ 0 & -1 & 1 \end{bmatrix} \begin{Bmatrix} \mathbf{u}_1 \\ \mathbf{u}_2 \\ \mathbf{u}_3 \end{Bmatrix}$$

In this form, K is singular or there is no one unique solution (the determinant is 0.0). In mechanics, the structure can move in space and hence any number of solutions could be obtained. To fix this problem, the matrix must be constrained.

$$\begin{Bmatrix} \mathbf{F}_1 \\ \mathbf{F}_2 \\ \mathbf{F}_3 \end{Bmatrix} = \begin{bmatrix} 2 & -2 & 0 \\ -2 & 2+1 & -1 \\ 0 & -1 & 1 \end{bmatrix} \begin{Bmatrix} 0 \\ \mathbf{u}_2 \\ \mathbf{u}_3 \end{Bmatrix}$$

The U_1 DOF is set to 0.0 (constrained) and this wipes out row 1 and column 1. This allows us to write the matrix in a more condensed form as shown below. The force at F_1 will be recovered later as a reaction force.

$$\begin{Bmatrix} \mathbf{F}_2 \\ \mathbf{F}_3 \end{Bmatrix} = \begin{bmatrix} 3 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} \mathbf{u}_2 \\ \mathbf{u}_3 \end{Bmatrix}$$

The determinant of the stiffness matrix is no longer zero (i.e., 2) and a solution can be found using any number of matrix technologies. Usually mathematicians will say that the matrix can be decomposed instead of inverted since the actually process is one of matrix segmentation and multiplication.

2011 – All Rights Reserved

Finite Element Technology

$$\begin{Bmatrix} 0 \\ 1 \end{Bmatrix} = \begin{bmatrix} 3 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix}$$

Starting from where we left off, we then can apply our loads. In this simple case, we are pulling on the end of the system with a force of 1.0. This would then imply the following: $F_2 = 0$ and $F_3 = 1.0$. We can then solve for our unknowns: u_2 and u_3:

$$\begin{Bmatrix} \mathbf{u}_2 \\ \mathbf{u}_3 \end{Bmatrix} = [\mathbf{K}]^{-1} \begin{Bmatrix} 0 \\ 1 \end{Bmatrix} \quad \textbf{and} \quad [\mathbf{K}]^{-1} = \frac{1}{2} \begin{bmatrix} 1 & 1 \\ 1 & 3 \end{bmatrix}$$

In this section, the matrix is transposed (which is really difficult with larger matrices) and we can solve for the unknowns.

$$\mathbf{u}_2 = \frac{1}{2} \longrightarrow \mathbf{F}_1 = -2 \bullet \mathbf{u}_2$$

$$\mathbf{u}_3 = \frac{3}{2} \longrightarrow \sigma = E \bullet \frac{\Delta u}{L}$$

Once displacements are generated it is not hard to then calculate the strain in each element, apply the elastic modulus and then calculate stresses.

*If the matrix is not constrained properly the solution will abort. Thus one of the most common error messages in NX Nastran has nothing to do with loads and everything to do with your constraints.

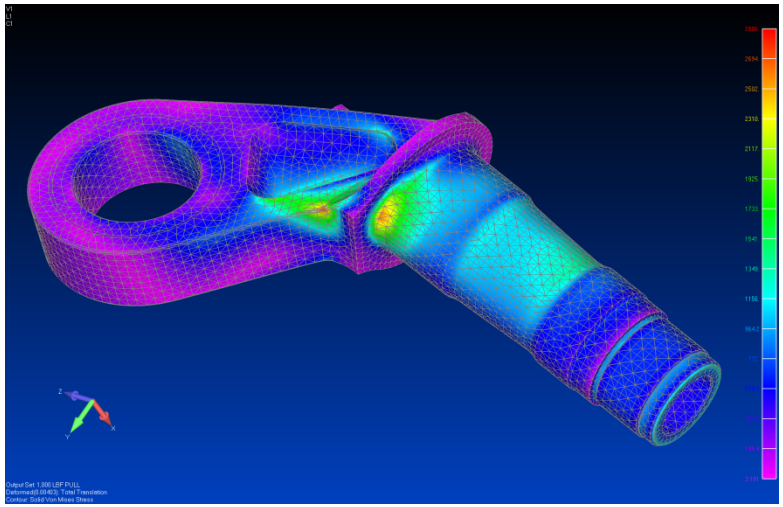
2011 – All Rights Reserved

Finite Element Technology

Workshop I: Introduction to Femap and NX Nastran

Pre-Processing Workflow:

- Walk through Interface. Introduce concept of Panes / Tool Bars / Menus
- Talk about Preferences and setting up one directory (Scratch) to store all of the modeling files.
- Femap is 100% Windows - Undo / Redo
- Import Geometry / Clean up Geometry using Geometry / Solid / Remove Face.
- Apply 1e4 load in –Z direction.
- Apply Constraints – Radial and Fixed.
- Analyze



Analysis Workflow:

- Geometry
- Material
- Property
- Mesh Sizing
- Meshing
- Loads
- Constraints
- Analyze

Note: All analysis examples in this class follows this general analysis outline.

Import Geometry File: Workshop 1 – Landing Gear Link / LANDING GEAR LINK.X_T

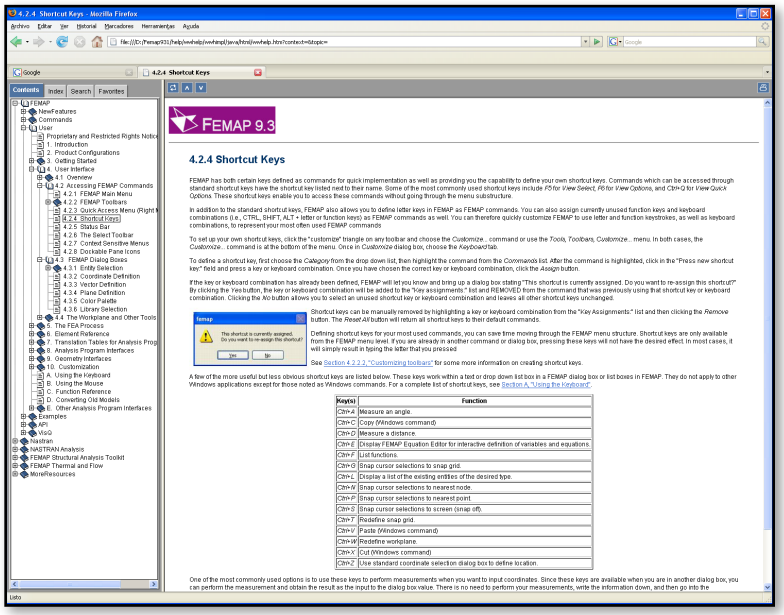
Movie File: Workshop 1 – Landing Gear Link / Workshop 1.wmv

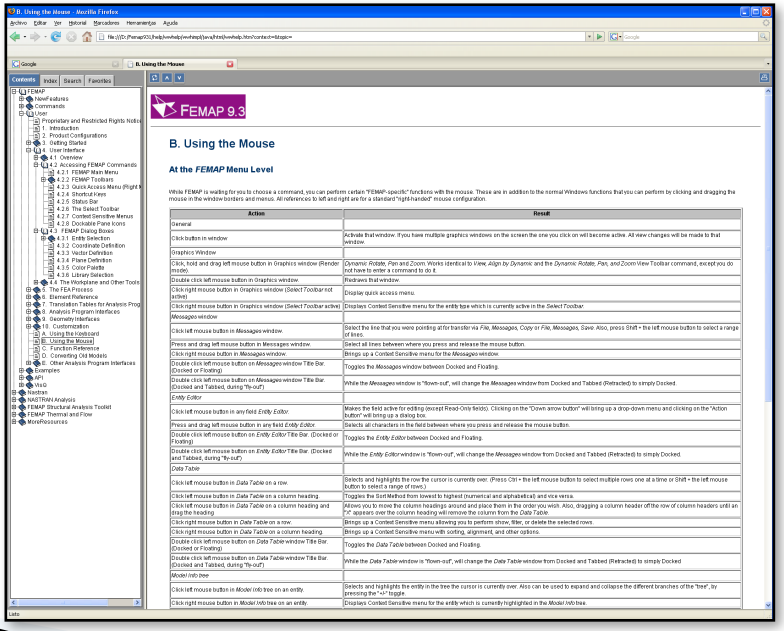
2011 – All Rights Reserved

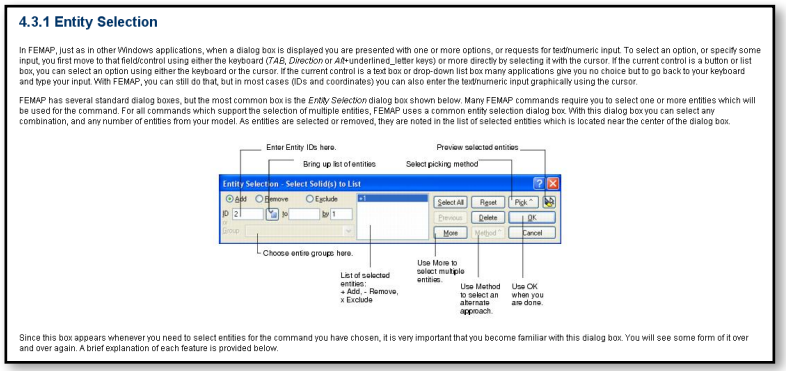
Femap Productivity Notes

Femap and NX Nastran Productivity (RTM)

- Reading the User manual can provide insight into how Femap functions.
- The complete set of NX Nastran PDF manuals are provided
- Explore online help under User Manual.
- Short cut key and Dialog boxes
- Look at “Using the Mouse”






4.3.1 Entity Selection
Enter Entity IDs here.
Preview selected entities
Bring up list of entities
Select picking method
Entity Selection - Select Solid(s) to List
Choose entire groups here.
List of selected entities:
+ Add, - Remove,
x Exclude
Use More to select multiple entities.
Use Method to select an alternate approach.
Use OK when you are done.

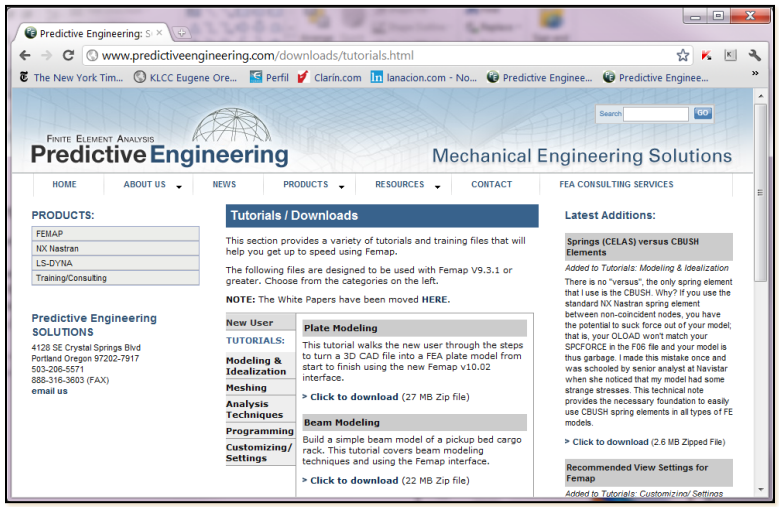
2011 – All Rights Reserved

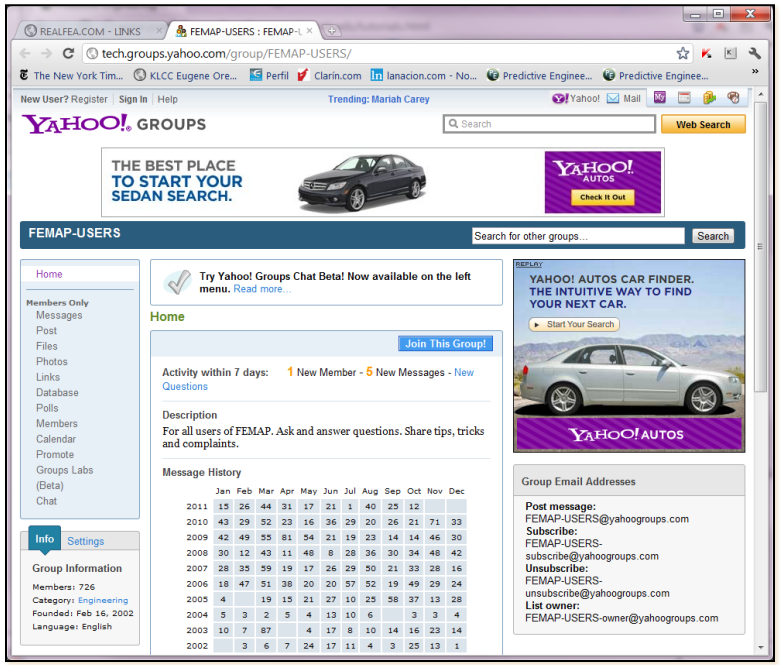
Foundations of FEA Modeling with Femap and NX Nastran

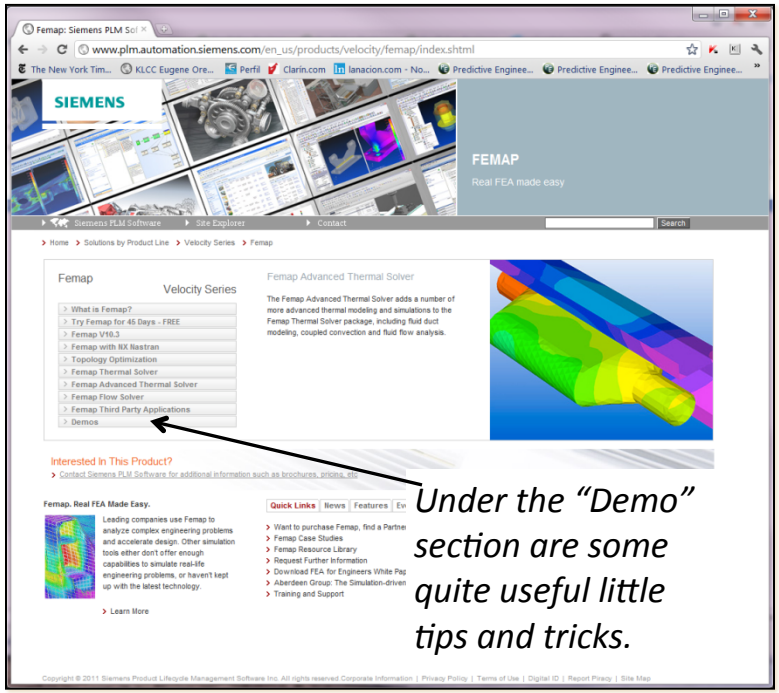
Femap Productivity Notes

Other Resources

- Extra Materials at the end of these class notes
- Predictive Engineering Website
- Siemens Web Site (under Velocity Series or www.Femap.com)
- Call Technical Support (Predictive Engineering or GTAC)
- Yahoo Groups has Femap-Users







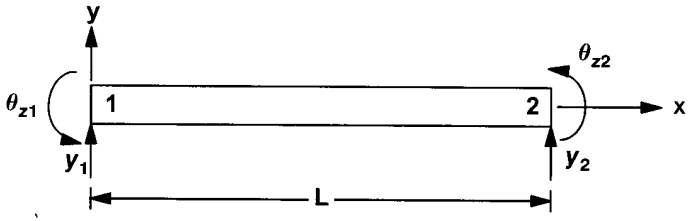
Under the "Demo" section are some quite useful little tips and tricks.

2011 – All Rights Reserved

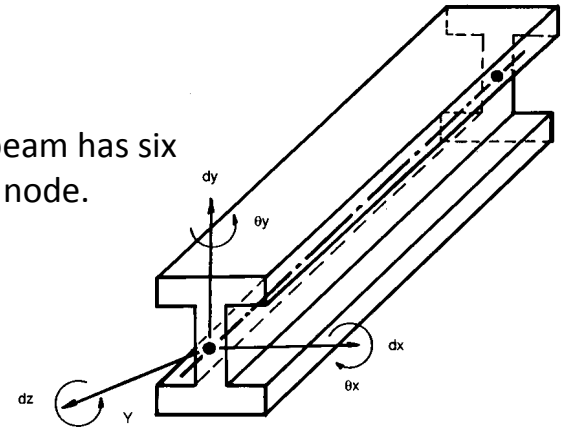
Finite Element Technology

Beam Elements: Nastran's Most Challenging Element

2D Beam model (for discussion):



A standard beam has six DOF at each node.



The equations for this 2D beam can be developed from straight mechanics (e.g., see Timoshenko). This is the fundamental concept about beams – they are exact elements.

$$\begin{Bmatrix} F_{y1} \\ M_{z1} \\ F_{y2} \\ M_{z1} \end{Bmatrix} = \frac{2EI}{L^3} \begin{bmatrix} 6 & 3L & -6 & 3L \\ 3L & 2L^2 & -3L & L^2 \\ -6 & -3L & 6 & -3L \\ 3L & L^2 & -3L & 2L^2 \end{bmatrix} \begin{Bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{Bmatrix}$$

You are a beam expert:

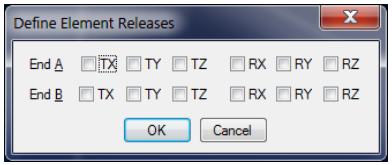
- *What dimension (rows/columns) would the matrix have for one beam element with two nodes?*
- *If you wanted to determine the stress at the base of a cantilevered beam having an end load, would you obtain better numbers by increasing the mesh density?*
- *How would you model a drilled hole in the web of your I-Beam shown above?*

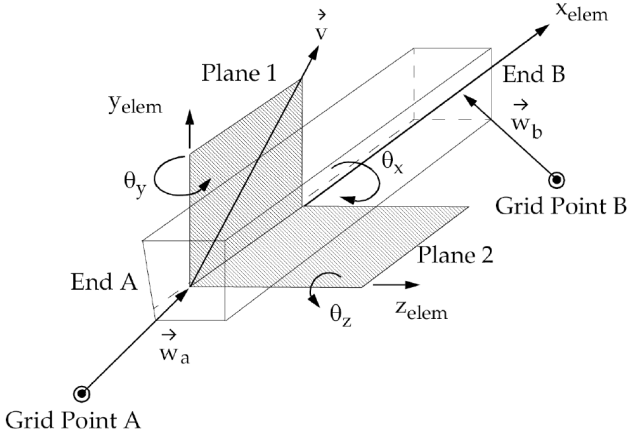
2011 – All Rights Reserved

Finite Element Technology

Basic Concepts on Working with Beams

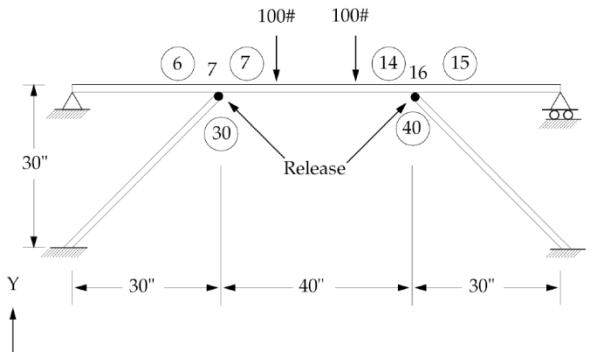
- Beams are used when you have long slender structural members with constant cross-sectional properties (e.g., structural steel tubing).
- One common rule-of-thumb is that the length to width is 10-1.
- An excellent review of beam nomenclature can be found in the NX Nastran's Element Library Reference (PDF file in the NX Nastran documentation).
- Beams have their own coordinate system. The beam's x-direction is down its length from End A (first node) to End B (second node).
- When meshed, the beam is always located at its shear center and is graphically drawn w.r.t. to this shear center.
- Beams can be offset to account for being attached to other thick structures (i.e., stiffened plates). Numerically a "rigid link" is used to offset the beam away from its neutral axis.
- DOF can be released at the beam ends to model a pinned joint or other types of behavior. (See Modify | Update Element | Beam/Bar Releases).





A schematic showing standard beam conventions for orientation and beam end offsets.

Pin flags (end releases) can be used for a variety of modeling tricks (e.g., a bridge pinned connection).

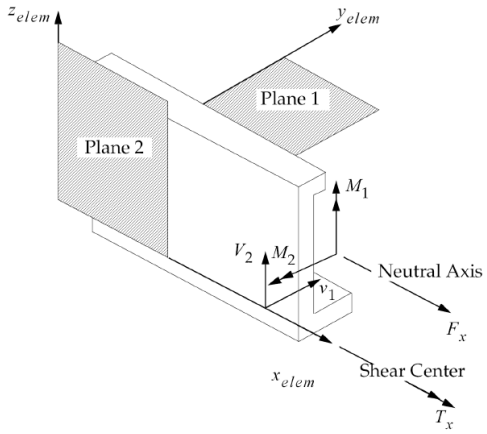


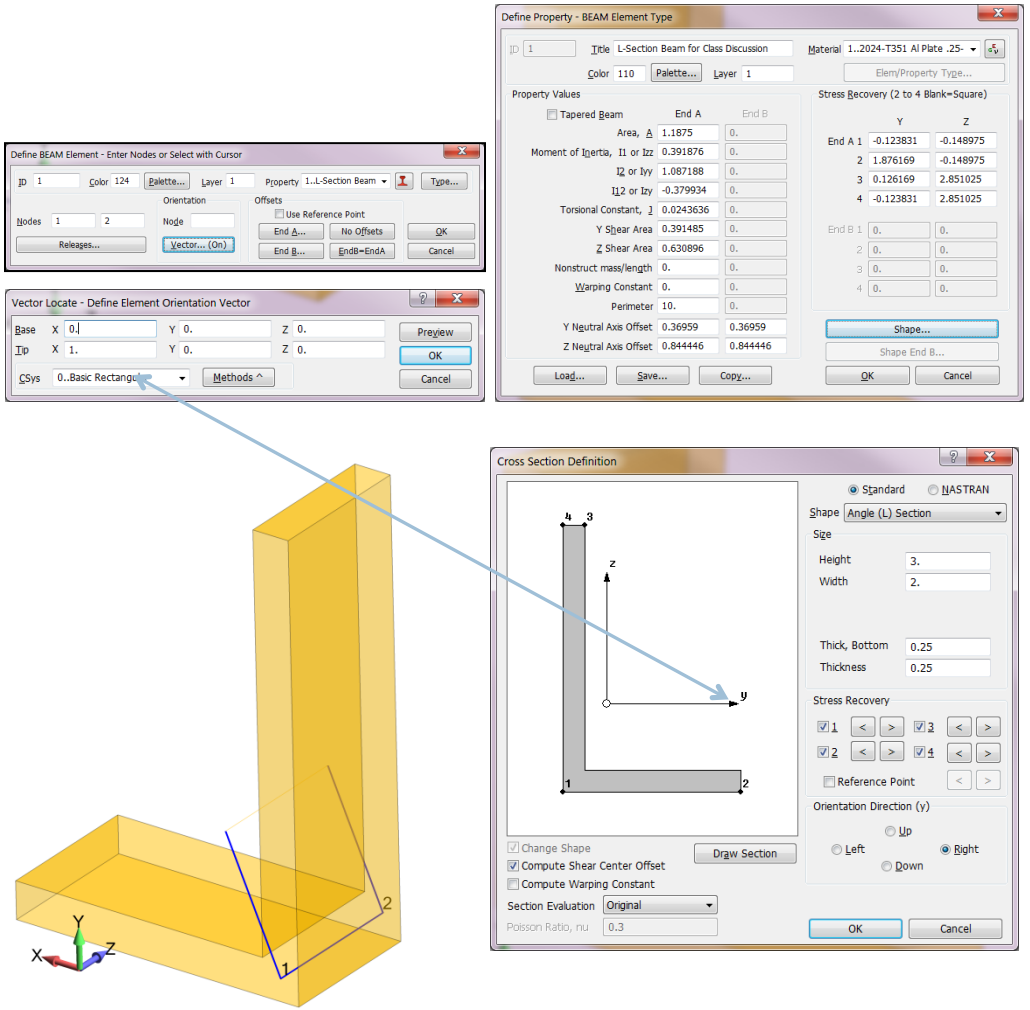
2011 – All Rights Reserved

Finite Element Technology

Basic Concepts on Working with Beams

- Beams need to be told how to orient themselves.
- The beam orientation is provided when you create the beam property. You define the Element Orientation Vector to match the beam's Y-axis.
- The nodes are placed by default at the beam's shear center; thus applied forces do not induce a twist or warping in the beam.
- The neutral axis is the center-of-gravity. Moments are calculated through the neutral axis.




Define BEAM Element - Enter Nodes or Select with Cursor
Vector Locate - Define Element Orientation Vector
Define Property - BEAM Element Type
Cross Section Definition

Smart and experienced engineers struggle with beam theory and its application. It is not exactly the most intuitive. When in doubt build a small pilot model to guide your investigation.

2011 – All Rights Reserved

Slide Reserved for Participating Predictive Engineering Clients

If you would like full access to these notes they can be purchased from Predictive Engineering.

Please contact: Training@PredictiveEngineering.com or call 503.206.5571

2011 – All Rights Reserved

Finite Element Technology

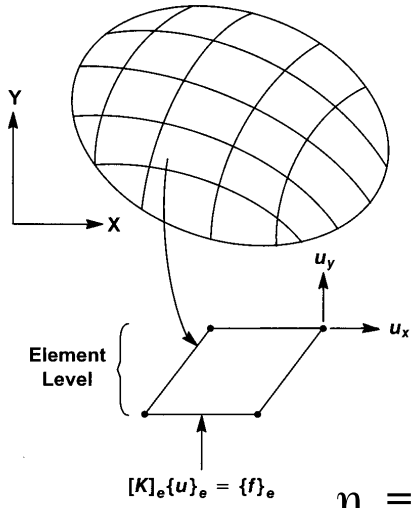
Isoparametric Elements: What Everybody Uses But Few Understand

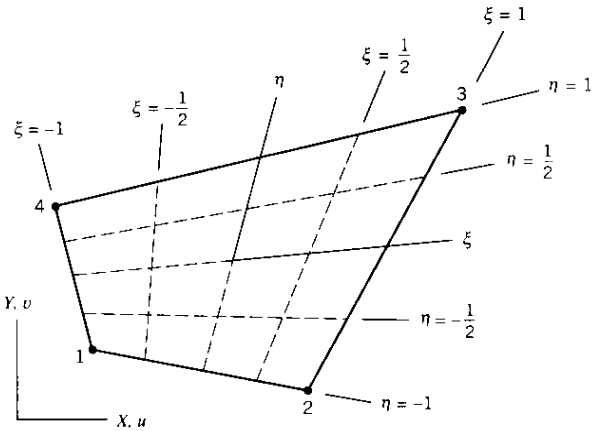
Isoparametric (having the same parameters under different coordinate systems) are the bedrock of modern FEA. Simple functions are used to discretize oddly shaped surfaces or volumes. The basis of this method is given in the subsequent slides. Although the theory is given in 2-D it can be directly leveraged into the third dimension.

One starts with a random region that is normalized into a -1 to +1 coordinate system and two formulas that use a simple linear shape function to define interior coordinates and interior displacements:

$$\mathbf{x_{xp}} = \sum_{\mathbf{i}=1}^{4} \mathbf{N_i}(\xi,\eta)\mathbf{x_{xi}}$$

$$\mathbf{u_{xp}} = \sum_{\mathbf{i}=1}^{4} \mathbf{N_i}(\xi,\eta)\mathbf{u_{xi}}$$





$\mathbf{N_i}$ is known as the shape function, which does double duty as the interpolation function for both coordinates (**x**) and displacements (**u**). This is the "iso" in the isoparametric. With these formulas we can map displacements in the interior of our element and also map any coordinates.

η = **eta**

ξ = **xi**

An example of a linear shape function for a four-node quadrilateral element (see FEA textbooks for quadratic shape functions use in parabolic eight-node quadrilateral elements):

$$N_1 = \frac{1}{4}(1-\xi)(1-\eta)$$

2011 – All Rights Reserved

Finite Element Technology

Isoparametric Elements: General Theory

We start with basic mechanics and apply the isoparametric method to these equations.

$$\sum \mathbf{F} = 0$$

Step 1: Satisfy static equilibrium

Step 2: Relate strain to displacements - simple 2D example

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \partial/\partial x & 0 \\ 0 & \partial/\partial y \\ \partial/\partial y & \partial/\partial x \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix} \quad \text{or} \quad \varepsilon = \partial \mathbf{u}$$

Step 2: Relate strain to displacements

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & \dots \\ 0 & N_1 & 0 & N_2 & \dots \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ . \end{Bmatrix} \quad \text{or} \quad \mathbf{u} = \mathbf{Nd}$$

This is where it gets a little complicated. To get our generalized displacements (**u, v**), the shape functions discussed on the prior slide are used to take corner point displacements (nodes) u_i and v_i and generate displacements anywhere within the element.

2011 – All Rights Reserved

Slide Reserved for Participating Predictive Engineering Clients

If you would like full access to these notes they can be purchased from Predictive Engineering.

Please contact: Training@PredictiveEngineering.com or call 503.206.5571

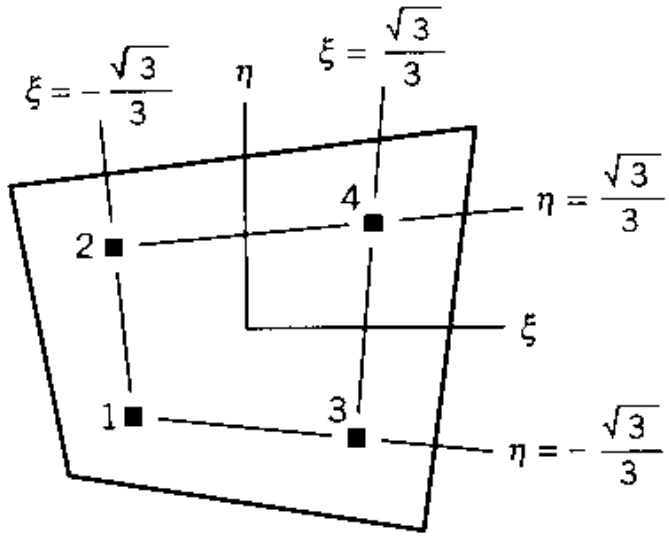
2011 – All Rights Reserved

Finite Element Technology

Isoparametric Elements: General Theory – Numerical Quadrature

To numerically integrate the Isoparametric element a technique known as Gauss Quadrature is employed. This technique is based on the element having a normalized coordinate system of -1, +1. Essentially, the inner terms of the stiffness equation given below are only solved at discrete points within the element and weighting functions based on Gauss Quadrature are then applied. The discrete points where this numerical integration is carried out are called "Gauss Points" or in long form, Guassian Integration Points. An example of the location of Guass Points in a quadrilateral element is given below.

$$\{k\} = \int_{-1}^{1}\int_{-1}^{1} [B]^T [E][B][J]\, d\xi\, d\eta$$



Guassian integration is at its best (i.e., most accurate) when the element is as near as possible to a perfect square. During the integration process, tabulated weighting values are used (terms Wi and Wj) to arrive at the final integrated value (I) for the elements area or volume:

$$I = \sum_{i=1}^{n}\sum_{j=1}^{m} W_i W_j \phi(\xi_i, \eta_j)$$

The location of these Guass Points are also used for strain recovery and with strain we have stress. That is, in Isoparametric elements, stresses are calculated at the Guass Points and extrapolated out to the nodal points for contouring. Hence, a high quality element (low Jacobian) will provide double benefits with a more accurate [K] and cleaner stress calculation.

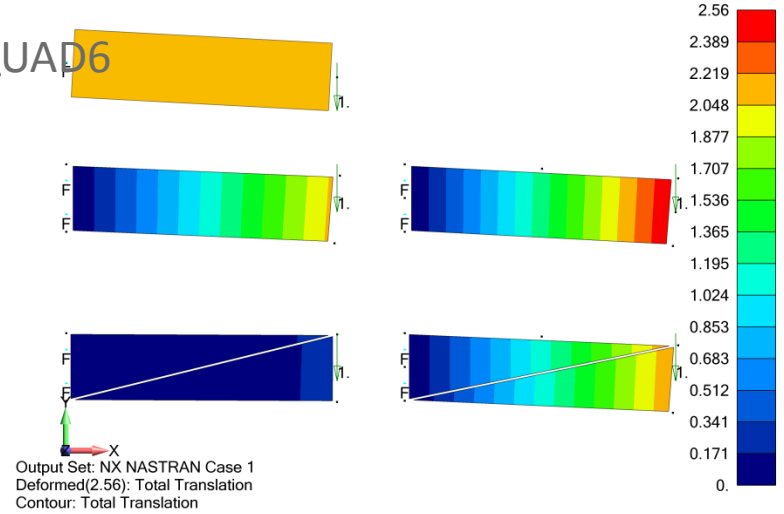
2011 – All Rights Reserved

Finite Element Technology

Isoparametric Convergence: Verification

Element Assessment: CQUAD4 | CTRIA3 | CQUAD8 | CQUAD6

The test models given on the right, represent an idealization of a cantilevered beam with a unit load at their ends. The theoretical displacement is 2.56. The beam element is exact while all other isoparametric formulations have values that approach this number to various degrees.



Student Tasks:

- Interrogated the model using the Selector Entity set to Node. Turn on the Entity Info Pane. Verify maximum node displacements for each plate formulation.
- Increase the mesh density of the beam element using Mesh > Remesh > Refine then reanalyze and check the results.
- Increase the mesh density of all the isoparametric elements by a factor of 2 using the Mesh Toolbox. (**Note:** Use Dialog Select within Meshing Toolbox to pick opposing curves to maintain quad mesh refinement.) Rerun the model and interrogate the results. Make note of relative numerical cost versus improved convergence.

Summary of Results

	Node	T2 Translation	DOF / Matrix
Beam	2	-2.56	12/144
CQUAD4	4	-2.02	24/576
CTRIA3	4	-0.19	12/144
CQUAD8	8	-2.51	48/2,304
CTRI6	9	-2.14	54/2,916

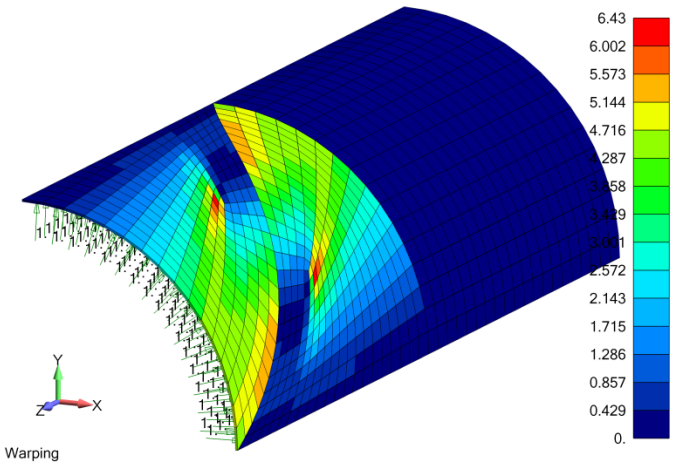
Model File: Instructors Models / Isoparametric Convergence / Isoparametric Element Comparison for Bending with One Beam Element - Start.modfem

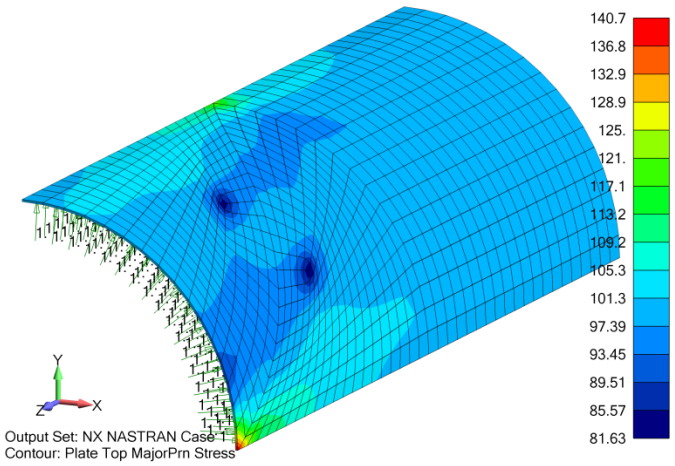
2011 – All Rights Reserved

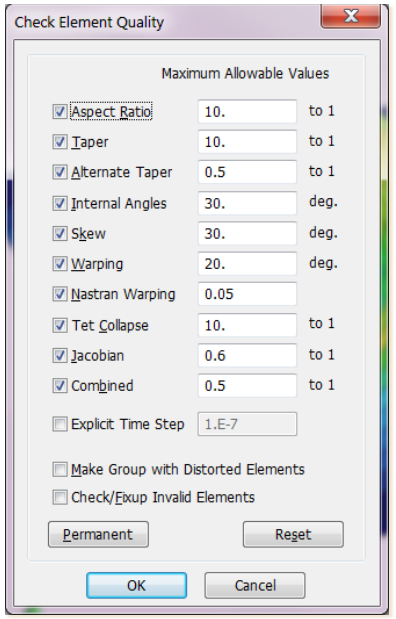
Finite Element Technology

Isoparametric Elements: CQUAD 4 versus CTRIA3 | Warping | Distortion

The Nastran CQUAD4 provides the accuracy of a eight-node CQUAD for the price of a four-node. However this trick of an embedded extra shape function only works for rectangular (2D) or hex (3D) elements. Triangular (2D) and tetrahedral (3D) elements remain linear and are well-known to be excessively stiff and can easily generate bad numbers. (Note: That is why the default tetrahedral element is the parabolic formulation with mid-side nodes.)
A warping element is shown below of a pressurized thin-shelled vessel. The hoop pressure should be a uniform 100 psi. In the mesh region with no warping and good element quality, the hoop stress is exactly 100 psi. Where the warping is the worst, the stress drops by 20%.







Open Model File: Instructor's Models / Plate Theory / Heavily Warped Cylindrical Model.modfem

Femap Command Manual: 7.4.5.6 Tools, Check, Element Quality

2011 – All Rights Reserved

Slide Reserved for Participating Predictive Engineering Clients

If you would like full access to these notes they can be purchased from Predictive Engineering.

Please contact: Training@PredictiveEngineering.com or call 503.206.5571

2011 – All Rights Reserved